Appendix II

SMPS Simulation with SPICE3

by Steven M. Sandler

Copyright 1997

pages 10-15

SMPS Simulation with SPICE 3

Steven M. Sandler

McGraw-Hill

New York San Francisco Washington, D.C. Auckland Bogotá Caracas Lisbon London Madrid Mexico City Milan Montreal New Delhi San Juan Singapore Sydney Tokyo Toronto

Library of Congress Cataloging-in-Publication Data

Sandler, Steven M.
SMPS simulation with SPICE 3 / Steven M. Sandler.
p. cm.
Includes bibliographical references and index.
ISBN 0-07-057853-2 (hardcover)
1. Switching power supplies—Computer simulation. 2. SPICE (Computer file). I. Title.
TK7881.15.S26 1997
621.381'044—dc21

96-37203
CIP

McGraw-Hill
A Division of The McGraw-Hill Companies

2 3 4 5 6 7 8 9 0 DOC/DOC 9 0 1 0 9 8 7

P/N 057853-2
PART OF
ISBN 0-07-913227-8

The sponsoring editor for this book was Steve Chapman, the editing supervisor was Marc Campbell, and the production supervisor was Don Schmidt.

Printed and bound by R. R. Donnelley & Sons Company.

McGraw-Hill books are available at special quantity discounts to use as premiums and sales promotions, or for use in corporate training programs. For more information, please write to the Director of Special Sales, McGraw-Hill, 11 West 19th Street, New York, NY 10011. Or contact your local bookstore.

This book is printed on recycled and acid-free paper containing a minimum of 50% recycled deinked fiber.

Smooth Transition switch, Von > Voff Case

```
.SUBCKT PSW1 1 2 3 4 {RON=1 ROFF=1MEG VON=1 VOFF=0}
*If VC > VON then RS=RON, If VC < VOFF then RS=ROFF,
*else RS 1MEG
B1 1 2 I=V(3,4) < {VOFF} ? V(1,2)/{ROFF} : V(3,4) > {VON} ?
+ V(1,2)/{RON} : V(1,2)/ (EXP(LN({RON*ROFF)^.5}) +
+ (3 * LN({RON/ROFF}) * (V(3,4) - {(VON+VOFF)/2})/
+ {2 * (VON-VOFF)}) - (2 * LN({RON/ROFF}) *
+ (V(3,4) - {(VON+VOFF)/2})^3 / {(VON-VOFF)^3} )))
.ENDS
```

Note: Pspice and Hspice do not support the Berkeley SPICE built-in S element. While hysteresis can be modeled with a subcircuit macro model in these programs, the approach requires several elements, is difficult to parameterize, and simulates very inefficiently when compared to the SPICE 3/IsSpice approach. In some cases, when the S element switch is used in a model as described in this book, the voltage controlled resistor, or the smooth transition switch may be substituted. Fig. 1.1 shows a simulation of the three different switches and their transfer functions.

Software Included with This Book

The diskette which is included with this book contains all of the models, circuits, schematics, and graphs that are used in this book. The schematics utilize the SpiceNet format. SpiceNet is a schematic entry program which has been specifically designed for use with the SPICE simulator. IntuScope is a post processor which is used to analyze SPICE output files.

The extensions correspond to the following file types:

.D1, CNT, LNK, C1—SpiceNet schematic files

.CIR—SPICE netlist for the top-level schematic diagram

.CKT—Full SPICE netlist which includes subcircuits and models

.OUT—IsSpice output file

.GA—IntuScope graph file

.LIB—SPICE Model Libraries

A evaluation version of ICAP/4, which includes demonstration versions of the SpiceNet, IsSpice, and IntuScope programs is available free of charge from Intusoft's web site, http://www.intusoft.com, and their CompuServe forum, CADD/CAM/CAE Vendor forum, Library 21.

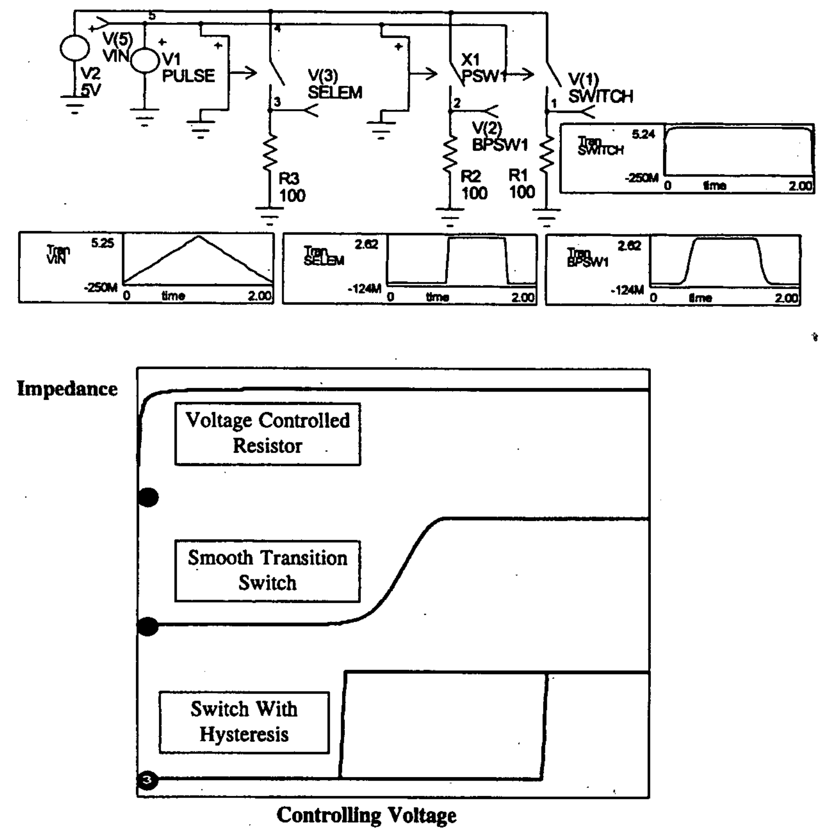

Fig. 1.1 The transfer function for the Berkeley SPICE 3 switch with hysteresis (selem), voltage controlled resistor (switch), and the B element smooth transition switch (PSW1).

SPICE Based Analyses Types Used in This Book

Operating Point Analysis

Produces the operating point of the circuit, including node voltages and voltage source currents.

The DC analysis determines the quiescent DC operating point of the circuit with inductors shorted and capacitors opened. A DC analysis, known as the "Initial Transient Solution", is automatically performed prior to a transient analysis in order to determine the transient initial conditions. A DC analysis, known as the "Small Signal Bias Solution", is performed prior to an AC small-signal analysis to determine the lin-

earized, small-signal models for all non-linear devices. It should be noted that these two operating point calculations can be different, depending on the DC and transient stimulus which is used.

Transfer Function Analysis

Produces a small signal DC transfer function.

The transfer function analysis calculates the small signal ratio of the output node to the input source, and also the input and output impedances of a circuit. This analysis may be used to determine the small signal gain and the input and output impedances of filter circuits. Any non-linear models, such as diodes or transistors, are first linearized based on the DC bias point, and then the small signal DC analysis is performed.

Sensitivity Analysis

Produces the DC and AC sensitivities of an output variable with respect to all circuit variables, including model parameters.

The sensitivity function uses the direct approach [35] to support sensitivity calculations for the DC and AC analyses. The DC sensitivity is with respect to the DC operating point. SPICE calculates the difference in an output variable, either a node voltage or a branch current, by perturbing each parameter of each device independently. Since the method is a numerical approximation, the results may demonstrate second order effects in highly sensitive components, or may fail to show very low but nonzero sensitivity. Since each variable is perturbed by a small fraction of its value, zero-valued parameters are not analyzed. This analysis is useful when trying to find a worst case scenario of circuit operation. By finding the most sensitive components and moving their values accordingly, the circuit's performance can then be evaluated.

DC Analysis

Produces a series of DC operating points by sweeping one independent source or two sources in a nested loop.

The DC analysis is used in applications which are dependent upon static variables such as line regulation, load regulation, or the DC modulation gain of a power converter. The .DC function is a special subset of the DC analysis feature. It is used to perform a series of DC operating points by sweeping voltage and/or current sources and

performing a DC operating point at each step value of the source(s). At each step, the DC voltages, currents, and computed device/model parameters can be recorded. The DC statement defines the sources which will be swept, and their corresponding increments. One or two sources can be involved in the DC sweep. If two sources are involved, the first source will be swept over its range for each value of the second source. This option is useful for obtaining semiconductor device output characteristics or calculating load lines.

AC Analysis

Generates a frequency response / Bode plot of the circuit. Magnitude, phase, real, or imaginary data is produced.

The AC analysis is used to evaluate many performance characteristics which are covered in this book. It may be used to determine performance characteristics such as circuit stability, impedance, and filter attenuation.

The AC analysis in SPICE computes the small signal response of the circuit. Output variables are recorded as a function of frequency. Before the AC analysis is performed, SPICE first computes the DC operating point of the circuit. It then determines the linearized, small-signal models for all of the non-linear devices in the circuit, based on this operating point. The resultant linear circuit is then analyzed over the specified range of frequencies. It is very important to establish the proper DC circuit biasing in order for the AC analysis to produce useful data. For example, biasing an op-amp in its linear range will give different AC results than if the op-amp is saturated.

Although the AC analysis performs a sinusoidal steady state analysis, it should not be confused with a transient (time domain) analysis using a large signal SINE wave. The AC analysis is a small signal analysis in which all non-linearities are linearized. For instance, if the DC biasing of a transistor gain stage produces a gain of ten, then the gain will remain ten, regardless of the input value. If the input is 1, then the output will be 10. If the input is 100, then the output will be 1000. The gain is linearized. Under non-linear conditions, however, the gain of the transistor will roll off as the input is increased. The "VName 1 0 SIN. . . ." stimulus is only used for non-linear time domain analyses, and should not be confused with the "Vname 1 0 AC 1" frequency response stimulus.

Frequency mixing note: The AC analysis is a single frequency analysis. Only one frequency is analyzed at a time. Therefore, circuits which perform signal mixing will not benefit from the AC analysis. In order to see frequency mixing, you will have to run a transient analysis and

convert the output waveforms into the frequency domain using a Fourier transform.

Transient Analysis

Runs a non-linear time domain simulation.

The transient analysis computes the circuit response as a function of time over any time interval. Output data, including node voltages and voltage source currents, can be recorded. During a transient analysis, numerous independent sources may have active time varying stimulus signals.

It is often necessary to start a SMPS simulation with a pre-defined set of operating conditions. The UIC (use initial conditions) keyword in the .TRAN statement causes SPICE to skip the initial transient solution (operating point) which is normally performed prior to the transient analysis. If this keyword is included, the values which are specified via "IC =" specifications on the various elements and .IC statements are used as the sole source for initial conditions. The transient analysis will begin with these values.

Fourier Analysis

Fourier analysis provides a simple means for evaluating the harmonic content of a time domain waveform. This analysis may be used to determine performance characteristics such as the conducted emissions performance of a switching power supply, or the harmonic content of a sine wave output converter. A Fourier analysis can be performed by SPICE, but is usually performed using a separate data post processing program which operates on the .PRINT transient simulation output data.

Temperature Analysis

SPICE allows the temperature of the circuit, or a particular element, to be varied.

SPICE simulates circuits using a global temperature of 27°C. This can be changed using the .OPTION TEMP= statement. In addition, SPICE 3 allows you to set the temperature for an individual device. This feature permits the simulation of a temperature gradient, as well as a "hot" device. Individual device temperatures are set directly on the device call line or in the .Model statement.

While the Monte Carlo, worst case, and optimization analyses are not inherently part of SPICE, most commercial vendors have added

them to the list of simulation capabilities. They are an invaluable part of SMPS investigation and design.

Monte Carlo and Worst Case Analysis

The Monte Carlo tolerance analysis is an ideal application for circuit simulation. The effects of component tolerance variations are difficult to assess by any other means. Imagine sitting in an engineering lab and sorting resistors, capacitors and other components in an attempt to find the worst case tolerance extremes to place in your circuit.

This investigation is usually performed either as a worst case analysis, or as a Monte Carlo Analysis. These analyses seem to be used interchangeably, although they are quite different.

A Worst Case analysis determines the worst case performance, but does not determine the statistical weighting of performance. As a general rule, the worst case analysis is preferred if the worst case values can be easily determined. In many cases, however, it is difficult to know which components must be varied, and in which direction, in order to generate the worst case result.

A Monte Carlo analysis provides the statistical weighting, but does not provide the worst case result. Monte Carlo analysis is generally used to calculate the mean and standard deviation of a particular performance characteristic. This analysis takes significantly longer to run than the worst case analysis, since it requires many simulations.

Optimizer Analysis

The Optimizer analysis is a powerful IsSpice feature that allows a series of simulations and measurements to be automatically performed over a range of component values. Either one or two circuit variables may be swept through a specified range of values. This feature is excellent for determining the damping components of an EMI filter.